How to use the laser cutter

Preparing the laser

1. Start the laser
Make sure that the key is inserted into the dashboard of the laser and that it is switched on. After the laser has been switched on, it will need a couple of minutes to boot up before it is ready. If the key is not there, the laser may be closed for the day. Contact the personnel at the workshop.

2. Turn on the ventilation system
It is imperative that the ventilation is running while you are using the laser. The switch can be found to the left of the computer. Turn the timer wheel a good amount to the right. If the ventilation is not on, or the laser produces an excessive amount of smoke, pause the laser and turn on the ventilation. When the ventilation has cleared the smoke in the lasering bay, you may restart the job.

NB! Make sure the workshop is not filled with too much smoke so the sprinkler system is not triggered.
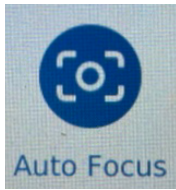
3. Insert the material
Place the material in the laser as far up and to the left as possible, as the laser cuts cleaner the closer it is to the upper lefthand corner. There are clamps found in a box and tape on a roll next to the lasers you can use to hold the material down. If the material has a bend to it, place the material so that the edges lift up from the table, so that you can use the clamps or tape to pin the material as flat as possible to the table. The thickness of a cut from the laser may vary from around 0.5 - 0.8 mm. The laser may cut or engrave materials as big as 1200mm 900mm. Optimal material dimensions for a clean cut is 800mmx600mm. You may **engrave any material thicknesses** and **cut the following thicknesses** with the **NTNU presets**:
- MDF: 3mm
- Thin birch plywood: 1mm & 2mm
- Plywood poplar: 4mm
- Grey cardboard: Up to 3mm
- Wood fiber cardboard: Up to 3mm
- Watercolor paper: Standard sizes
- Acrylics: Up to 9.5mm

NB!
- **Do not use used acrylics as they may catch on fire. Also make sure that the material you are using is acrylics and not polycarbonate, as lasering polycarbonate is prohibited.**
- **Do not move or lift up the grid table, as it must be positioned properly to not crash into the walls during autofocus.**

4. Focus the laser

The autofocus button can be found in the dashboard of the laser. Before using the autofocus, make sure to position the laser using the joystick on the dashboard so that it is parked directly above your material, and that the material is pinned as flat as possible. The manual focus tab must also be fixed in the upright position. When pressing the autofocus button, the table will rise until the material makes contact with the laser, then lower a bit.

If the autofocus is broken, contact the staff. You can then focus the laser manually. First flip down the manual focus pin, and adjust the height of the table by using the buttons on the dashboard until the pin is barely making contact with the material. When this is done, the laser will be in focus. Be sure to fix the pin in the upright position afterwards. If the height of the table is adjusted after the laser is focused, you will have to focus the laser again. If the laser is out of focus, the cut may not go all the way through, and the laser will produce an excessive amount of smoke.
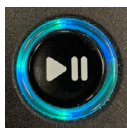
Manual focus tap in upright and deployed position.
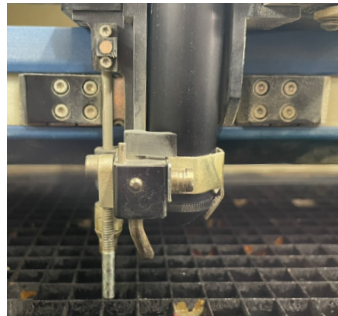
5. General use of the dashboard

Emergency stop button
To be pressed in case of an emergency, such as fire or excessive smoke.

Start/stop button
Used to start a lasering job. If pressed once during a job, the laser will pause after having completed the current line it is working on, if pressed again, it will cancel the job.

The joystick
Moves the laser manually when you are in the joystick-tab on the dashboard.

Outline/trace
Traces the outline of a job, so that you can make sure it is within the bounds of the material, and that none of the clamps are in the way.

The laserpointer
Makes the laser display a small red dot on the material directly below it.

Delete your jobs after use
Make sure to delete your jobs on the laser when you have finished using it.This way, it will be a better work environment for all:)

The settings under the settings tab are allready calibrated, and must not be touched.

Preparing the file

1. Prepare the file before coming to the workshop
Always try to prepare your file before coming to the workshop to do your cutting. This way, you avoid unnecessary stress and the cutting sessions become more effective, and more people get to use the laser. We recommend using Rhino for making your lasering files as you may sometimes encounter bugs when using Archicad DWG files.

2. Upload your file to the computer
Use a memory stick to upload your file to the computer. You may also log into your cloud service if necessary, but make sure to log out properly when you are finished with your work. Save the file in the downloads tab, as it makes it easier for the staff to keep the computer running smoothly and the file structure clean. When you know you are finished using the file for lasering, make sure to delete it from the computer. All files on the computer get deleted on regular intervals, so it may not be possible to keep working on the same file the next time you use the laser.

3. Delete superfluous lines
When engraving, small scale details may overflow into each other so that the area becomes excessively dark, or the detail becomes unreadable. Delete or simplify such lines for a better result. If you are going to cut or engrave an intricate pattern or something similar with many intersecting lines, joining these lines will save you a lot of time. An example of this is brick patterns. It is also wise to delete any duplicate lines from your file. In Rhino, this can be done with the command "Seldup", and then deleting any selections.

4. Material effective layout
First select material plates adequate for your lasering job. Do not use bigger plates than necessary. After this, create a square representing the area of your materials. Then, position each shape you wish to cut out as closely together as possible in the upper left hand corner. This saves materials and time as the laser will have to move less around. Each shape needs to be no more than 5mm apart. Place each shape so that the side which needs to be the prettiest is facing up, as the laser produces more burn marks on the underside of the material. These can be sandpapered away if needed.

5. Layering
Divide lines you want to be cut or engraved into different colored layers. You can also subdivide these further into different layers if you want specific lines to be engraved or cut before others. The software differentiates between different layers by using the colors assigned to them. It is always a good idea to engrave before cutting, as parts which are cut loose may move a bit. Always use vector engraving when engraving anything else than an image, as this saves time.

Sending file to print from a program to epilog manager

When you have properly prepared the file, you want to print and placed your material on the laser properly, you are ready to send your file to print. To get a smooth printing process, choose different colors for cutting and engraving. The laser automatically cuts and engraves in its own chosen programmed order, which to us is not necessarily logical.

There are two ways of sending your file to print:
1. Selection method: Highlight what you want to print, press Ctrl+P, and a dialog box will appear.
2. Window selection method: Press Ctrl+P first, then choose "Window" and drag a selection box around your design. When asked about scaling, click "Yes" and adjust the scale afterward.

Important settings:

- Ensure the printer is set to Vector Output (for cuts; verify if Raster is needed for engraving).
- In the print window, confirm the scale—never use "Scale to Fit." Instead, manually scale your file in rhino and set the scale to 1:1 in the print-window. You may also scale the file in the print-window, though this may sometimes lead to bugs.
- The preview (right-side white box) should display your design. If it's missing, recheck the scale or resend the file.

When everything looks correct, click Print.

Fitting the laser file to the laser bed

The laser's dashboard will now show your material. Often, the file appears outside the visible area. Drag a large selection box to locate it. Once found, position it over your material. If scaling and measurements were done correctly, this should go smooth.

The next steps:
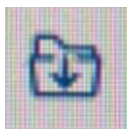
1. Click "Split by Color" in the right-hand menu.
2. Set all layers to Vector (not Engrave).
3. Use the icon to load preset settings (e.g., "3mm MDF Cut" and so on). We've saved profiles for each material, including cut and engrave options.
4. Reorder layers by dragging—the topmost layer processes first.
5. Finally, click "Send to Print" (bottom-right corner).

Printing and running the laser

Your file will now appear in the laser's menu. Select it and click **"Trace"** to verify placement and check the estimated job time. Once confirmed, press the **Play/pause button** to start. You can pause anytime by hitting the same button.

Reminder:
- Never leave the laser unattended.
- Avoid leaning on the machine—it can misalign the laser.
- Do not move or lift up the grid table as it has to pe positioned properly so it wont crash into the walls during auto focus.
- If you see excessive smoke or flames, pause immediately and notify Lucas Workshop staff or student assistants.
- Always delete your file when you are finished with the laser.

Typical errors that can occur while lasering:

- Print does not arrive to the machine - Just try again.:)
- Rhino license not found – Make sure you are using Rhino 8. If this does not solve the issue, check if the machine is connected to the internet. You may need to restart the computer and give it some time to reboot before trying again.
- Trace does not show the correct selected file – We then recommend to send it once again to the laser machine. If it is still wrong, try to press play and see if it cuts the right file anyway. Sometimes there is just a delay in the system. You could also try to delete other jobs listed below your selected file.
- Trouble with a PDF? - PDF files can cause problems. Often, lines do not join correctly. We always recommend using .dwg or .ai if possible.
- The laser refuses to engrave a hatch? – Try putting the hatch in a vector square/frame. Contact the staff if this issue persists.

Happy lasering and good luck gluing<3